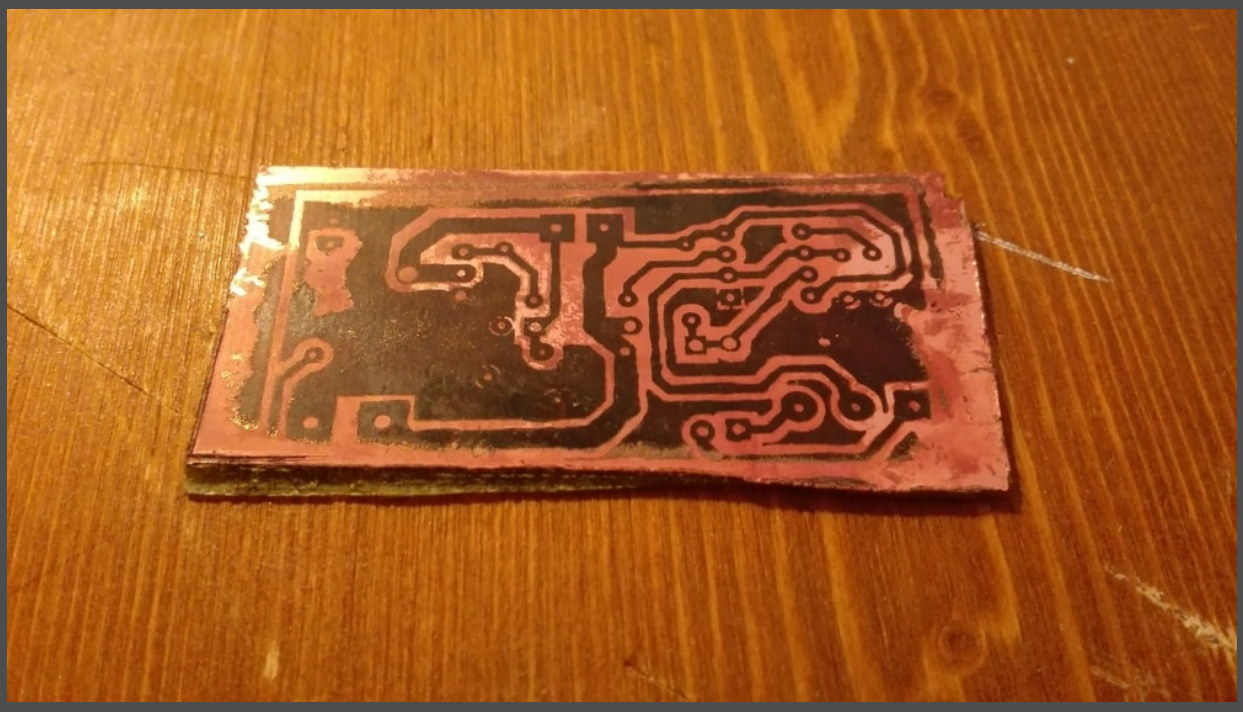


Print PCB (direct transfer method)

Preface

In electrical engineering, the production of printed circuit boards has changed over the last decades and has produced various techniques. Since the professional production of boards for small quantities in technical processes is still very cost-intensive, hobby electronics use their own manufacturing processes for prototypes. Thus, the direct transfer method is one of the more well-known process techniques, which is considered in more detail in this project documentation.

The publication of the last documentation revealed that the functionality of the hand drill had to be extended. A controller is developed that controls the number of rotational speed of a motor. The circuit diagram and the circuit board drafts of a finished project were downloaded. **Spoiler**: The project was stopped after several attempts, since the procedure was defined after detailed consideration as cumbersome, environmentally harmful and cost-intensive (see conclusion). This documentation is not a tutorial, but a project termination report (see DIN 69901).

Safety instructions Ferric Iron (III) chloride 40%

Ferric chloride 40% is a very toxic chemical that should not be used without safety clothing. This includes gloves and safety glasses. Children should not work with this chemical. It is also better to wear old clothes that can leave some stains. The chemical can cause skin irritation and severe eye damage. A workroom should always be well ventilated and there should be no pets nearby. It is very important that you read the safety instructions on the packaging very carefully.

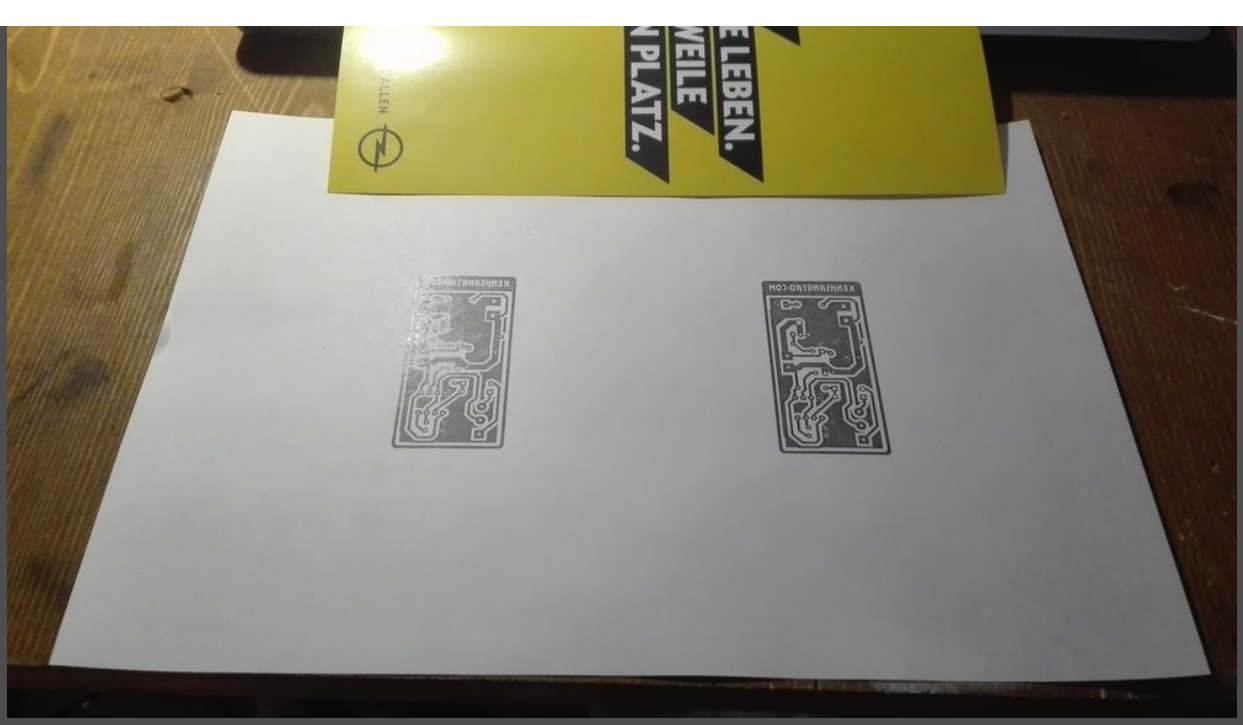
Materials

The materials include not only various chemicals, but also much that can be found in the household. You have to pay attention to the safety of your body when working and therefore you have to pay attention to personal protective clothing. Safety glasses and latex gloves should at least be provided. In addition, also a workplace and not necessarily the precious wood table of the grandparents. Please also note that plastic bowls (e.g. Tupper) can no longer be used for food.

- Iron (III) chloride 40%
- Latex Gloves
- Safety Goggles
- Toilet Paper
- Noil Polish Remover (with Acetone)
- Steel Sponge
- Dishwashing Liquid
- Plastic Trais
- Printer (Dell C1760)
- Copper blanks (blanks/unbored/untreated)
- Catalogue pages or similarly coated paper
- Crepp Tape
- Clothes Iron (Severin BA 3206)
- Carpet knife or metal saw
- Set Square

Documentation

In order to be able to work more precisely, a test print of the downloaded files is created in order to determine the position as precisely as possible. A standard A4 sheet of paper from the printer is used for this purpose.



It is possible to hold the test print (on the white sheet of paper) and the catalogue page in the backlight. With this you can see if the position fits later.

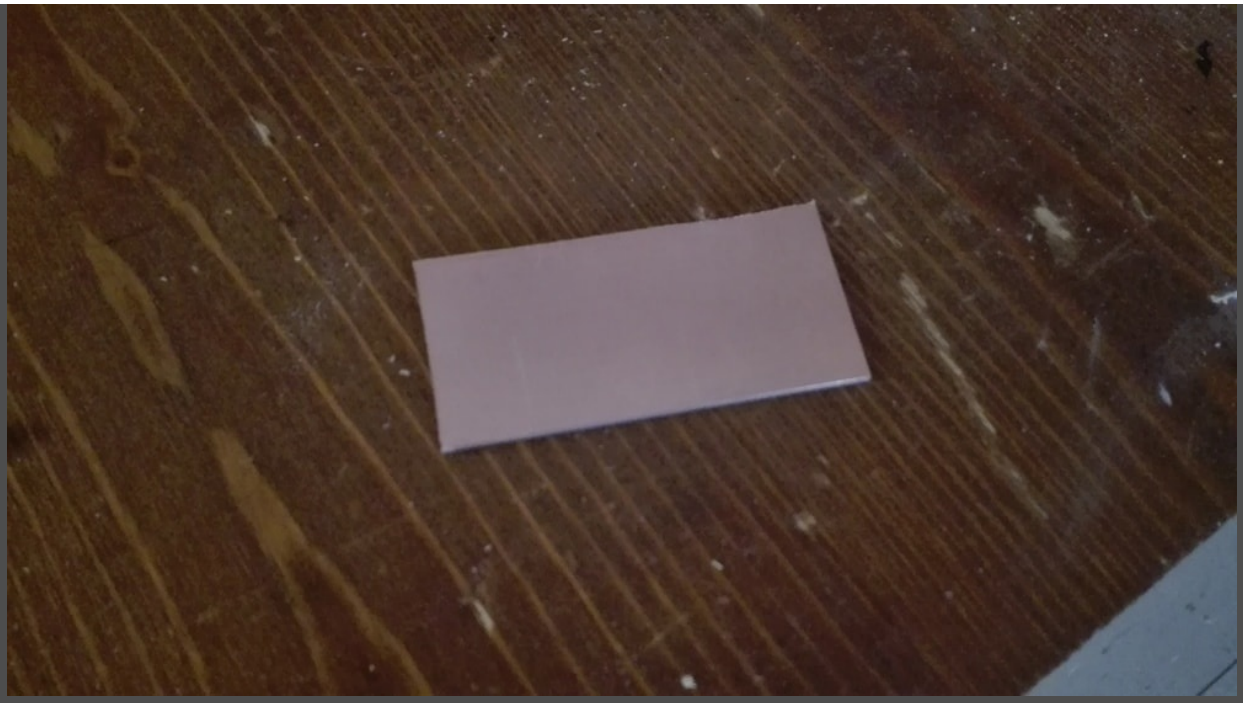
I needed almost ten tries before the imprint fitted. On some pages, for example, the coating was too thick and the ink couldn't hold. Also the catalogue pages were too small for the printer and I had to glue them on a white sheet of paper. But that didn't work very well and so I simply adjusted the printer with the tracks better.

Here you can see the almost perfect print.

So that we do not waste too much material from the prototype PCB, we place the print on the edge of the PCB to measure the size. At this point you can decide for yourself how neatly you want to work. Since this was only an attempt for me, I worked fast.

There are several ways to cut the prototype PCB. In this documentation I cut the plate with a carpet knife. This is very exhausting, takes a long time and the result gets messy. You can do this better with a hacksaw, because the edges get straighter. If you have the possibility, and you have a model table circular saw, you should use it. This will produce the best results and everything will be very neat.

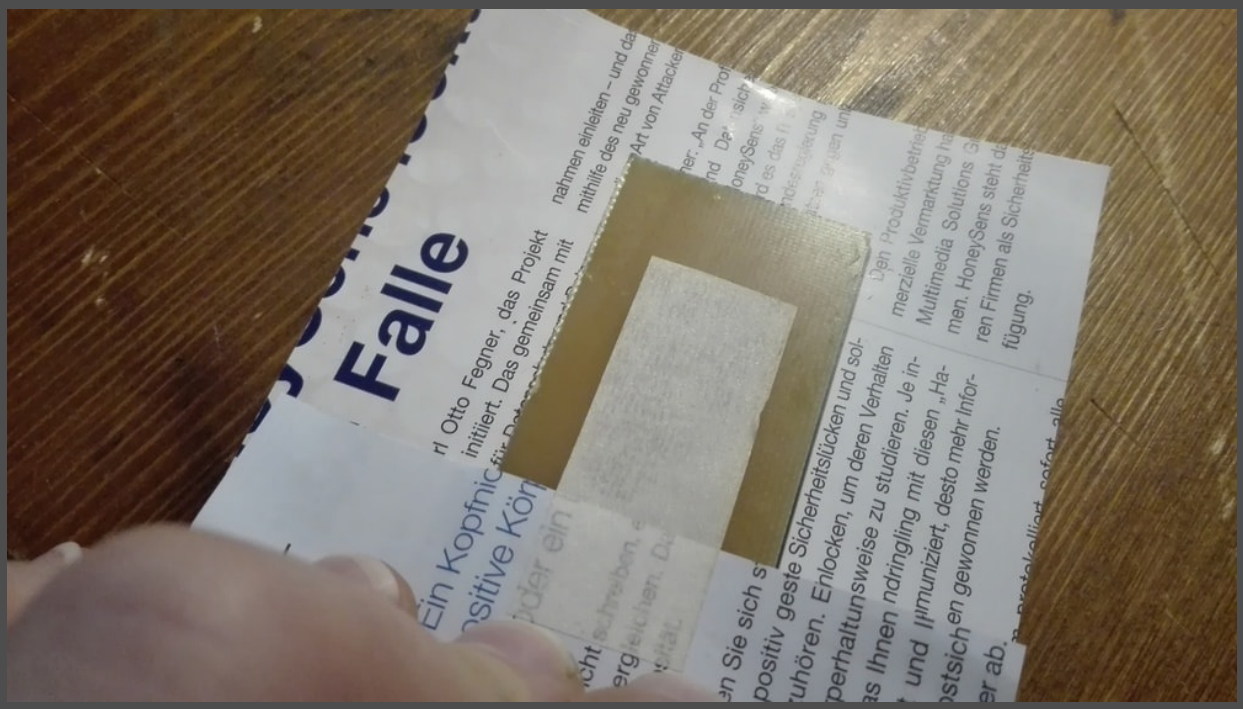
Here you can see the finished piece of the prototypes PCB.



The next step is to clean the plate. For this we use nail polish remover with Acetone, because this dissolves fat and residues best. Please make sure that the working area is well ventilated, because Acentone can cause dizziness after a few minutes. Pour a little of the remover onto a piece of toilet paper. This can be done very quickly, but the tidier you work on this spot, the better the result will be. As you can see from my result on the first photo, I worked much too messy.

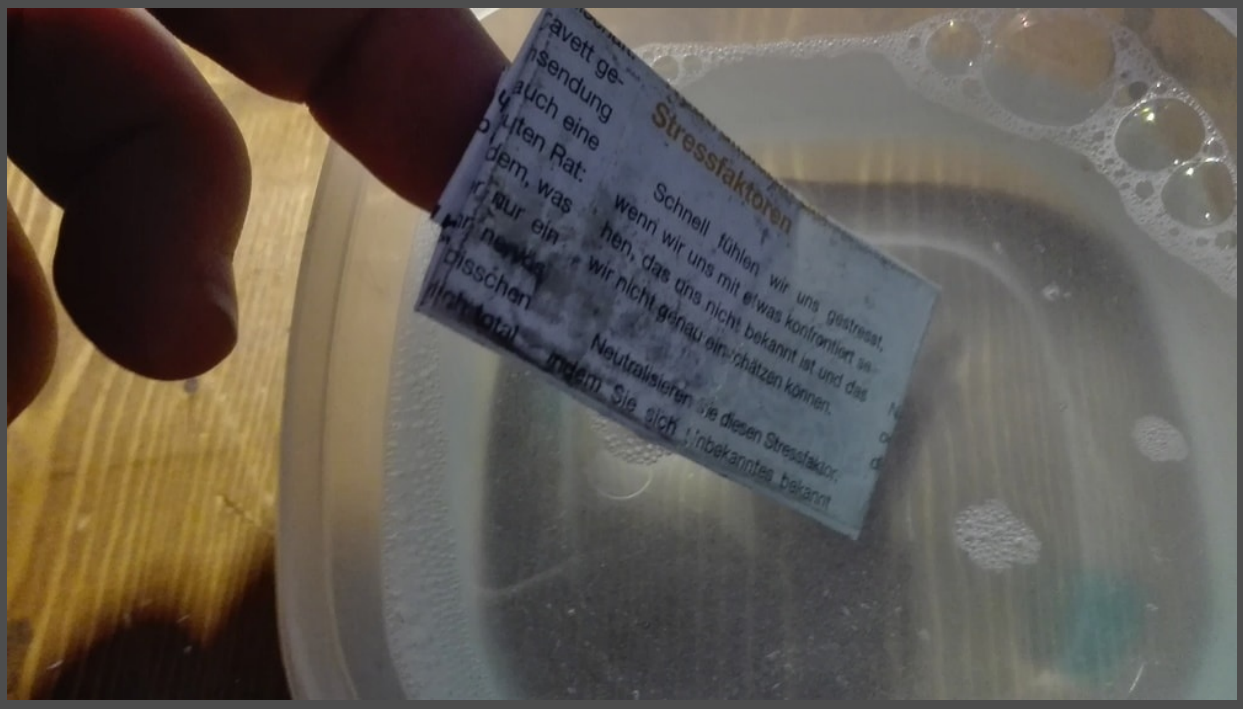
Now we cut a piece of catalogue paper to size. This should be the size as you can see on the photo. You need a 3-4 cm thick edge. Then you put the prototypes PCB with the copper side on the print. This allows the print to be transferred from the catalogue side to the copper side of the PCB.

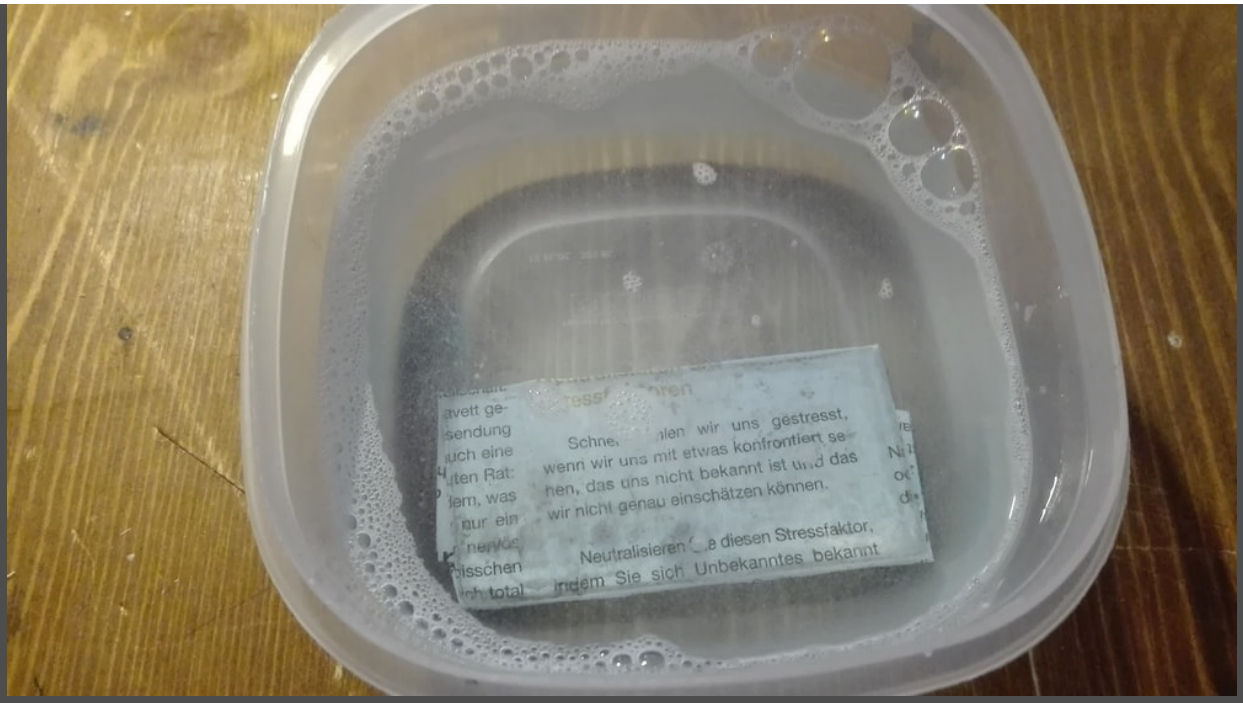
So that everything holds well during ironing, we now pack the protoytp PCB. Each side is turned over and stuck with a small piece of crepe tape.



The copper plate inserted in the catalogue page is ironed in several passes. The duration of the process is influenced by the performance of the iron, the quality of the printing ink and the catalogue page, etc. The copper plate is ironed in several passes. Unfortunately I can't give an exact time here how long you should iron. With me it was apparently already too long, because parts of the imprint are a little blurred.

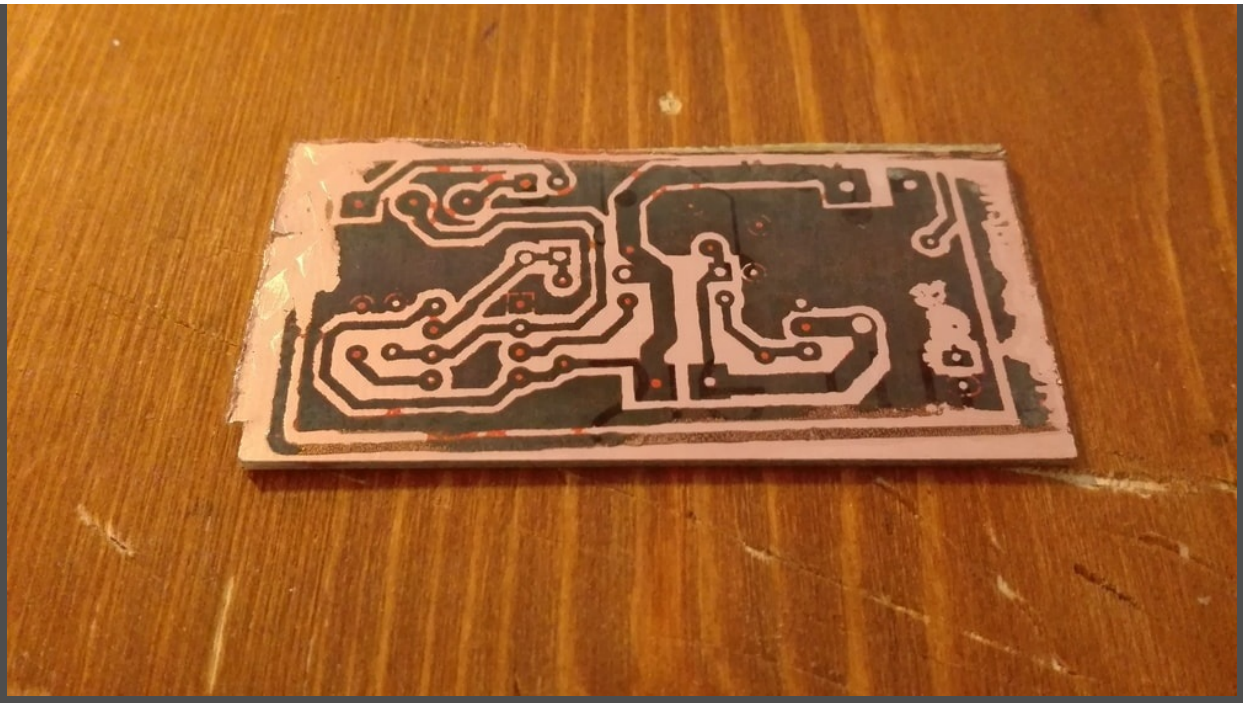
We then place the PCB packed in the catalogue page in a dish with soapy water. It must remain there for at least 30 minutes so that the firmly pressed paper can be removed without damaging the imprint.





Here you can already see that the imprint was not as neat as I would have liked it to be. The process is very difficult and you need a lot of experience with the chemicals and materials. Anyway, I thought it would be much easier.

It doesn't look like a homemade electronic board, but rather like abstract art.



If you etch the circuit board in the iron chloride you should not use metal pliers as the chemicals attack and damage the metal. There are very good plastic pliers that can be ordered on the Internet. At the latest here one should really wear the safety glasses, so that one does not get chemicals in the eyes. The concentration of the Einsenchlorid determines how long one must leave the circuit board in the bath. You can take it out and have a look. Since I was impatient my result looks bad. To clean the board, we scrub the surface with a steel sponge. With it we remove the remaining black imprint. Then you can drill and solder the board. I have omitted that here, since this board is never inserted anywhere.

Workflow Notes

- There are far too many factors that can cause the project to fail or produce poor quality results. The catalogue page, the (printer) blackness, the temperature of the etching liquid, for example, and all in all the effort does not appear to be justified in any way in relation to the result. The quality of the end product also depends on the working material used.
- The excessive use of chemicals, such as ferric chloride with its sludge formation. Nail polish remover with its narcotic effect caused by acetone and other cleaning agents does not exactly contribute to the protection of the environment and one's own health. Chemistry, as a natural science, has more than once triggered a revolution in the history of human development, but the etching of copper plates must be viewed critically once again.
- The waste of material is not insignificant. Catalogue paper, printer cartridges, crepe tape, toilet paper etc. accumulate if you want to produce more than one printed circuit board. Even if these costs can be reduced e.g. by the marker method or etching with a cutting plotter, the costs for the above mentioned chemical products still remain.

- The process techniques studied so far are not suitable for beginners, since they require an extraordinary chemical and technical knowledge and one must gain at least a few months of experience in the manufacturing process, which is only achieved by continuous exercises.

Conclusion

There will be follow-up activities for this project, as research on further process technologies will have to be intensified. A solution will be sought which reduces or revises the negative points mentioned above. The project has failed in this form and the experiences will be taken over in the following documentations. Etching printed circuit boards using the direct toner method is no longer a recommended technique.

Unfortunately I haven't found a better solution yet. My experiments with cut copper tape are interesting, but not enough for professional projects. A printed circuit board has to last longer than a week. The methods with copper tape fit much better if you work with fabric, fashion or works of art. Since I bought the chemicals, I will use them anyway. But I won't buy any new ferric chloride in the future. There are some other ways to make PCBs and I will investigate these methods as well. With this method I am really very unhappy and that is not even due to my more than bad result.

What many critics forget at this point is that this is exactly how science works. One has heard or read about a method and copied it. Then one notices that the method is shit and improves the method and/or the process engineering. You make some mistakes and learn a lot. At some point you are satisfied with your method so far that you make your results available to the general public. At some point another person comes to your method, notices that it all sucks and improves your method. This is how electrical engineering works. That's how science works. Therefore, all scientific methods should be available to the general public.